

Improved Modeling Capabilities in Glenn-HT – The NASA Glenn Research Center General Multi-Block Navier-Stokes Heat Transfer Code

**J. D. Heidmann
NASA Glenn Research Center
Cleveland, OH**

**A. A. Ameri
U. of Toledo @ NASA Glenn Research Center
Cleveland, OH**

**V. K. Garg
U. of Toledo @ NASA Glenn Research Center
Cleveland, OH**

**D. L. Rigby
QSS, Inc. @ NASA Glenn Research Center
Cleveland, OH**

ABSTRACT

The NASA Glenn Research Center Multi-Block Navier-Stokes Heat Transfer Code, Glenn-HT, has been developed for and applied to a wide variety of turbine convective heat transfer problems. These problems have included tip clearance flows, internal cooling passage flows, and external turbine blade flows, including film cooling. The code has been validated against experimental data for a wide variety of turbine heat transfer flows. The general multi-block capability of the code makes it useful for computations of complicated three-dimensional flowfields in turbines as well as other propulsion system flowfields where convective heat transfer is important. The code is able to accurately predict wall heat transfer through a combination of detailed boundary layer resolution and advanced modeling capabilities.

Recent code development has concentrated on improving the code's predictive capability and extending its usefulness to a broader range of flow problems. Conjugate heat transfer capability has been incorporated into the code using the Boundary Element Method to allow simultaneous computation of fluid and solid heat transfer without requiring a solid volume grid. This capability is being extended to layered solids, such as a turbine blade with a thermal barrier coating, as well as to solids with variable thermal conductivity. Reynolds Stress turbulence modeling efforts have been underway to improve predictions through the incorporation of anisotropic effects. Automatic topology generation techniques are being developed to shorten the calculation cycle through automation of the gridding process. This is particularly important for extremely complicated geometries, such as the cooling passages inside turbine blades and vanes, which can require thousands of grid blocks. Efforts are underway to incorporate unsteady flow capability into the code. This may be useful for studying the transient heat transfer phenomena associated with turbine accelerator missions under the Access to Space Program. These and future improvements to the Glenn-HT code are aimed at improved accuracy, speed, and flexibility of the code for future convective heat transfer issues in turbines and other propulsion systems.

INTRODUCTION

Modern gas turbine engines operate at very high temperatures due to the improved cycle efficiencies afforded by such temperatures. This is especially important for upcoming designs in the Next Generation Launch Technology (NGLT) Program. For example, the Turbine-Based Combined Cycle (TBCC) Engine Project, under NGLT, requires extremely high temperatures in the turbine and diffuser to meet the cycle requirements. Such high temperatures must be met with a minimum of cooling, since the heat sink available to the engine is limited by the high temperature of the compressor discharge air and high cooling requirements of other vehicle systems. In order for these requirements to be met, accurate and timely analysis of the convective heat transfer is very important. The analysis must be performed early in the design cycle to allow incorporation of the results into actual designs. The Glenn-HT code is being developed with these requirements in mind.

The Glenn-HT code had its genesis in the TRAF code of Arnone et al. [1]. The TRAF code was a 3-D Navier Stokes flow solver for external flow over turbine vanes and blades. The code allowed for non-matching grid lines in the wake region of a turbine blade or vane for improved grid orthogonality. Several improvements were made to the code by Ameri and Arnone [2] to incorporate near-wall heat transfer modeling features, such as a two-equation turbulence model and emphasis on the use of adequate grid resolution near the wall to capture turbine convective heat transfer accurately.

The next major improvement was the extension of the code from a single-block grid to general multi-block capability by Steinthorsson et al. [3]. This allowed the code to be applied to geometries of greater complexity than before, such as complicated film and internally cooled turbine blades that had become of interest in the turbine heat transfer community. The general multi-block capability is also very important for the complex geometries which must be modeled in the TBCC project, such as cooled turbine rear frame, centerbody, and diffuser liner geometries.

Applications of the Glenn-HT multi-block code have included turbine tip clearance calculations of Ameri et al. [4, 5], turbine film cooling analyses of Garg et al. [6, 7] and Heidmann et al. [8], and turbine internal coolant passage simulations of Rigby et al. [9, 10]. Recently, the code has been applied to other convective heat transfer problems not associated with turbines, such as aircraft wing internal anti-icing and turbine-based combined cycle diffuser flows. All of these applications were enabled by the multi-block capability of the code. In addition, the heat transfer predictions of the Glenn-HT code in these cases were in good agreement with experimental data, validating the heat transfer modeling of the code for propulsion system convective heat transfer.

In the past few years, Glenn-HT code development work has continued on several fronts at NASA Glenn Research Center. Dr. James Heidmann has been developing a conjugate heat transfer capability in the Glenn-HT code in cooperation with Dr. Alain Kassab and Dr. Eduardo Divo of the University of Central Florida. The Boundary Element Method (BEM) of Kassab et al. [11] is used as a subroutine in the Glenn-HT code. This enables the Glenn-HT code to predict not only the temperatures in the flow field, but also in the bounding solid. This will allow code users to better predict the real flow environment. Dr. Ali Ameri has been incorporating higher order turbulence models into the code. These models, including the Reynolds Stress and $\bar{\nu}^2 f$ models [12], allow for turbulence anisotropy through the use of a greater number of modeling equations. Modeling anisotropy of turbulence is considered important in many complex flows, especially those found in highly three-dimensional propulsion flows. Finally, Dr. David Rigby has been developing an automatic topology generation system called TopMaker [13]. TopMaker will automatically create a multi-block topology for a given geometry. Once the topology is created, standard grid generators can create the grid. The development of this capability will greatly speed the grid generation process, since topology creation is currently a manual process which can be very slow for complicated geometries.

The use of the Glenn-HT code for the geometries of interest to the propulsion heat transfer community depends upon its continued development. In addition to the three areas of development described above, an updated version of Glenn-HT has been completed by Dr. Erlendur Steinthorsson using the improved object-oriented capabilities of Fortran90/95. This version of the code will be more

easily able to incorporate these and future development efforts, as well as allowing for more general flow capabilities such as the ability to model unsteady flows. This report will describe the development efforts and indicate the areas of potential application for the newest version of the Glenn-HT convective heat transfer code.

CODE DESCRIPTION

The Glenn-HT code is a general purpose three-dimensional flow solver designed for computation of convective heat transfer of flows in complicated geometries. The code solves the full compressible Navier-Stokes equations using a multi-stage Runge-Kutta-based multigrid method. The finite volume method is used with central differencing, and artificial dissipation is employed. The overall accuracy of the code is second order. The present version of the code employs the $k-\omega$ turbulence model developed by Wilcox [14, 15], with modifications by Menter [16] as implemented by Chima [17]. Accurate heat transfer predictions are possible with the code because the model integrates to the wall and no wall functions are used. Rather, the computational grid is generated to be sufficiently fine near walls to produce a y^+ value of less than 1.0 at the first grid point away from the wall. A turbulent Prandtl number of 0.9 is used and laminar viscosity is determined from temperature using a 0.7 power law [18]. Specific heats are assumed to be constant. A full description of the code and its recent applications to turbine heat transfer can be found in [19].

CONJUGATE HEAT TRANSFER

The traditional method for analyzing the heat transfer on a turbine blade or other convective surface is to first obtain a fluid-side convection solution assuming either isothermal or constant heat flux conditions at the blade surface. This effectively decouples the fluid solution from the thermal conduction inside the blade material. For a one-temperature problem (e.g., one without film cooling), the external flow solution is used to compute a heat transfer coefficient distribution on the surface. For a two-temperature problem (such as one with film cooling), a second external flow solution is obtained using a different wall thermal boundary condition. The two solutions are then used to compute the heat transfer coefficient and film effectiveness distributions. The current Glenn-HT code allows the user to specify any distribution of wall temperature and/or heat flux over the convective surface. However, the specification is fixed, and so is not affected by conduction in the solid. The conjugate method allows for a coupled heat transfer solution between the solid and fluid, and thereby more accurate heat transfer predictions. Accounting for conjugate heat transfer is particularly important where there are large thermal gradients on the surface, such as might be found near film cooling holes or other geometrical complexities.

The boundary element method (BEM) computes the temperature in a solid using discretization on the surface itself – not in the volume of the solid. This is enabled by the fact that the conduction of heat in a solid is governed by the Laplace equation for temperature. The Laplace equation is converted into a boundary integral equation (BIE), which then can be solved on the surface. Once the BIE is solved, the temperature at any point within the volume of the solid can be easily determined. A complete description of the BEM technique is available in [11]. The major advantage of BEM for a coupled fluid/solid solution is that the convective flow solver already has a surface discretization, so no further grid generation is necessary. This greatly reduces the time required to generate a conjugate solution versus the traditional method of volumetric discretization of the solid.

Figures 1 and 2 show the wall temperature and heat flux, respectively, for the leading edge region of a Honeywell film-cooled Inconel turbine vane [20]. Figure 1 shows that the highest wall temperatures occur in the showerhead region, between rows of holes. The effect of heating on the inner plenum wall can also be observed. For an adiabatic wall case, the inner plenum wall temperature would remain near 0.5. Figure 2 indicates that high heat flux into the fluid occurs near the shaped film cooling holes on the vane pressure side and inside the angled film cooling holes, due to streamwise thermal conduction and accelerating flow, respectively. Heat fluxes into the solid are observed in the showerhead region. Results such as these indicate that the unified approach to conjugate heat transfer afforded by the Glenn-HT/BEM coupling may lead to improved heat transfer designs without greatly adding to the computational cycle time.

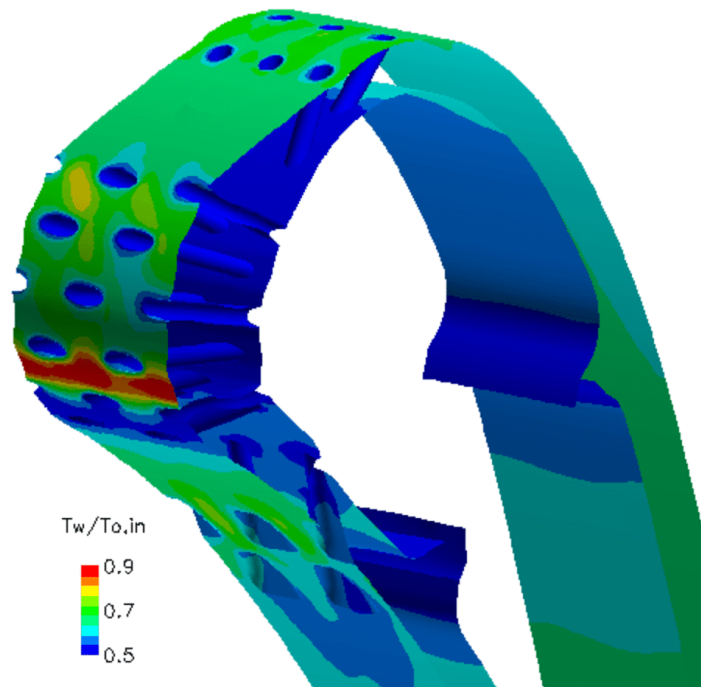


Figure 1: Conjugate wall temperature prediction for Honeywell Inconel film-cooled vane

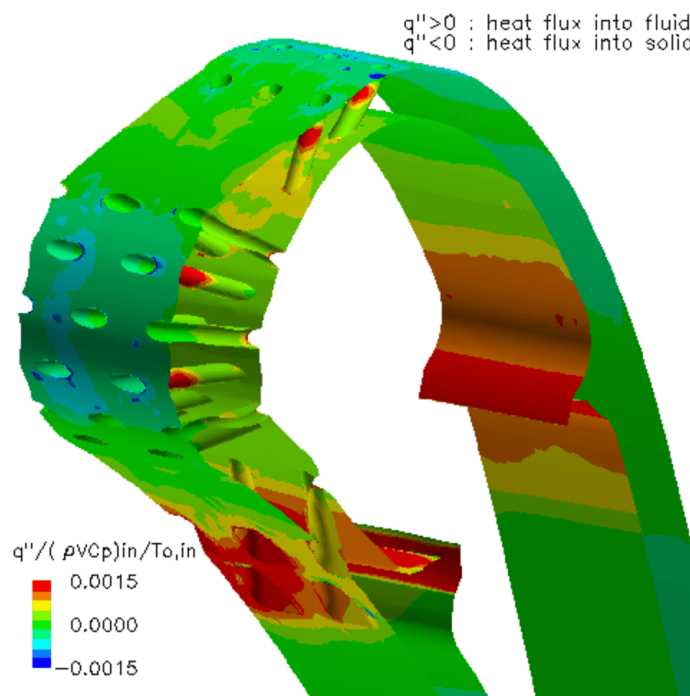


Figure 2: Conjugate wall heat flux prediction for Honeywell Inconel film-cooled vane

TURBULENCE MODELING

The standard method of modeling turbulence as used in CFD codes is the use of two equation eddy viscosity models which utilize transport equations for a velocity scale and a time scale. The most common types are $k-\epsilon$ and $k-W$ models. Turbulence modeling is often faulted as the cause for deviations from measured data in prediction of blade heat transfer. Turbulence models are designed to predict shear flows and often do predict the correct level of heat transfer for such flows. Prediction of blade heat transfer however includes at least prediction of stagnation region heat transfer as well as endwall and near-endwall and the near-tip and the tip heat transfer. This involves complex flow fields which are different from the simpler shear flows. More sophisticated models which do not use the eddy viscosity hypothesis and directly model the Reynolds stresses are good candidates for use. Unfortunately these models which solve transport equations for Reynolds stresses and an equation for ϵ are CPU consuming and difficult to converge.

In recent years a new Reynolds stress model which is based on W equation in place of ϵ has been devised by Wilcox [14] which has very good numerical properties. This model was implemented in Glenn-HT and was used to calculate the heat transfer over the Transonic Cascade of Giel et al. [21]. Another type of model, still using the eddy-viscosity hypothesis, is the \bar{v}^2-f model of Durbin [12]. It has shown promise for prediction of heat transfer on turbine blades. The \bar{v}^2-f model is a 4 equation model. The Glenn-HT code was fitted with this model to run in parallel for a multi-block structured grid topology. A comparison of these two new models to the ones that are more popularly used was carried out. The calculations were done using a grid similar to the multi-block grid shown in Figure 3. The grid contains approximately 800,000 nodes and is refined near the walls to capture heat transfer. This is done by choosing the non-dimensional distance (y^+) at the first grid point away from the wall to be near unity. Figure 4 shows the mid span results obtained for the transonic cascade blade heat transfer using a simple algebraic model as well as the Shear Stress Transport (SST) model of Menter [22] as performed by Garg and Ameri [23], the Stress- W model and the \bar{v}^2-f model. The SST model is an amalgam of the $k-\epsilon$ and the $k-W$ model and itself is a two-equation model. It is obvious from that figure that the Stress- W model overpredicts the heat transfer on the pressure side and greatly over-predicts the heat transfer in the stagnation area. The best results are obtained using the \bar{v}^2-f model.

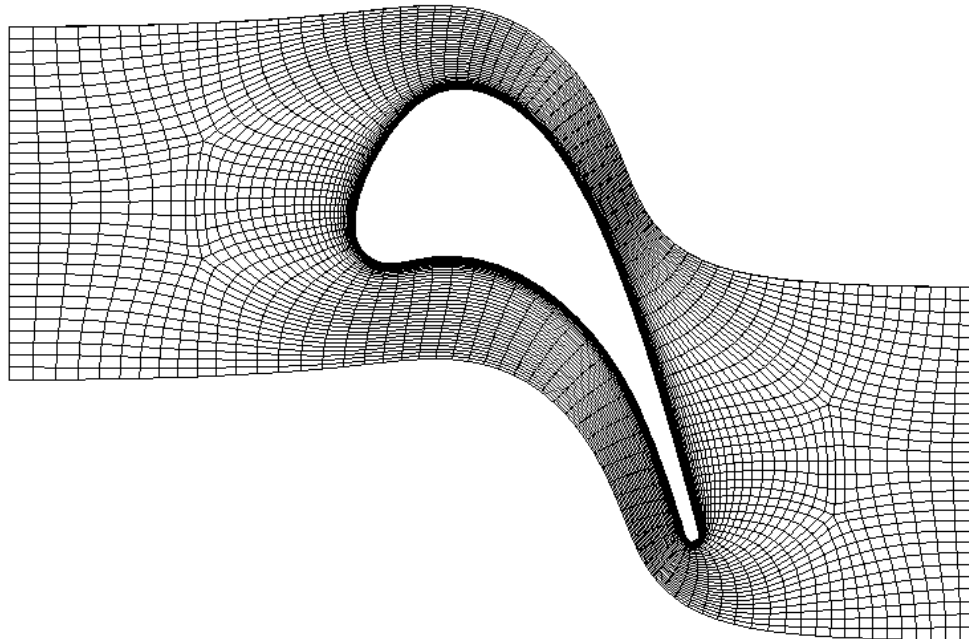


Figure 3: Computational grid for NASA Glenn Research Center transonic cascade

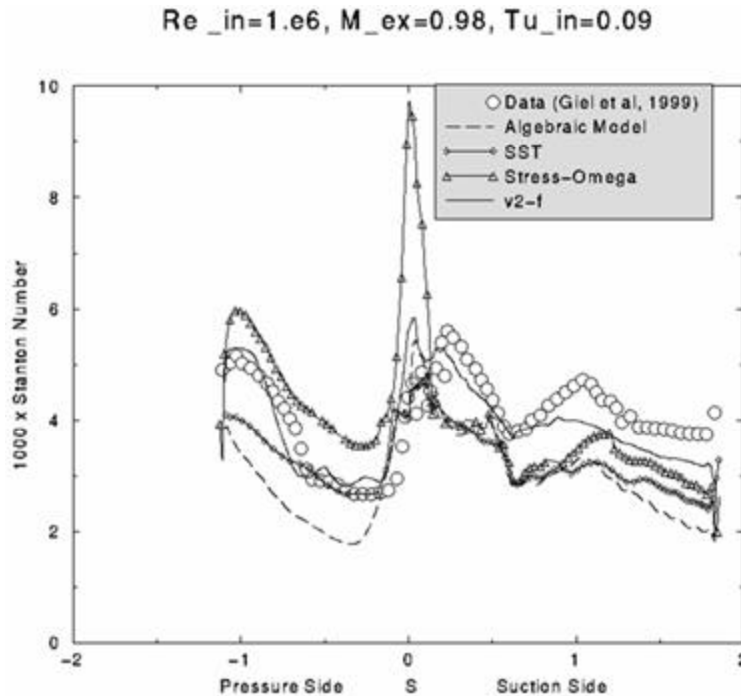


Figure 4: Computed heat transfer for transonic cascade for various turbulence models

AUTOMATIC TOPOLOGY GENERATION

In recent years, the Glenn-HT code has been applied to flows in progressively more complex geometries. For example, several calculations performed by NASA Glenn researchers under the Ultra-Efficient Engine Technology (UEET) Program simulated the flow through the complex internal coolant passages inside real turbine blade designs. These calculations involved Pratt & Whitney, General Electric Aircraft Engines, and Honeywell. The grids required to model these very complicated flow passages required up to 10 million grid points per case. In addition, the clock time required to solve a given problem has decreased due to increased CPU speed and greater parallelization. These two factors have caused the percentage of real time spent for a given problem on grid generation to increase. It is not unusual for the generation of a high-quality viscous multi-block structured grid for a complex geometry to require months for a single researcher to produce. Because of this, it is very difficult to incorporate analysis results into the design cycle.

In an effort to greatly reduce the time required to generate grids for complicated geometries, development of an automatic topology generation system called TopMaker [13] is under development at NASA Glenn Research Center. Since the Glenn-HT code uses structured multi-block grids, a topology must first be generated which defines the eight corners of each block and how they are connected. Once this topology is generated, it is a relatively straightforward process to produce a grid. Currently, the researcher must generate the topology by hand, which can be very time-consuming for a complex geometry. However, TopMaker offers the capability to automatically produce a valid topology given only the geometric definition of the flowfield.

TopMaker uses the medial axis of the geometry. The medial axis in two dimensions is defined as the collection of points that are equidistant to at least two locations on the boundary [24]. In two-dimensional space, medial vertices occur at locations that are equidistant to at least three locations on the boundary. Using the medial axis and vertices, along with the "touch points" which are the points on the surface which are equidistant from the medial vertices, a valid topology is produced by following relatively simple rules for the various types of vertices and edges which are possible. Among the features

that are desirable for viscous grids is that clustered grids remain attached to solid boundaries, and that orthogonality of the grid is maximized. Figure 5 shows the progression from medial vertices and edges to the final topology for a simple example in two-dimensions. Figure 6 shows the final topology for an anti-icing geometry inside an aircraft wing. Again, this topology is not the computational grid itself, but rather defines the corners of each grid block. A grid generator such as GridPro™ [25] can be used with the topology to quickly generate the final grid.

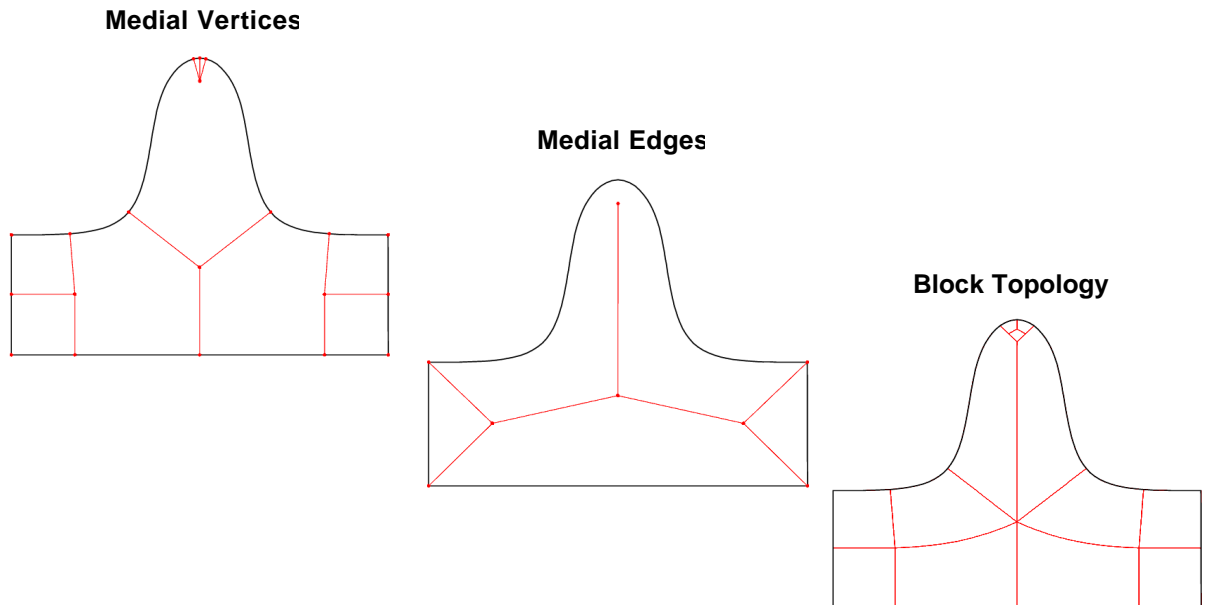


Figure 5: Progression from medial vertices and edges to final block topology

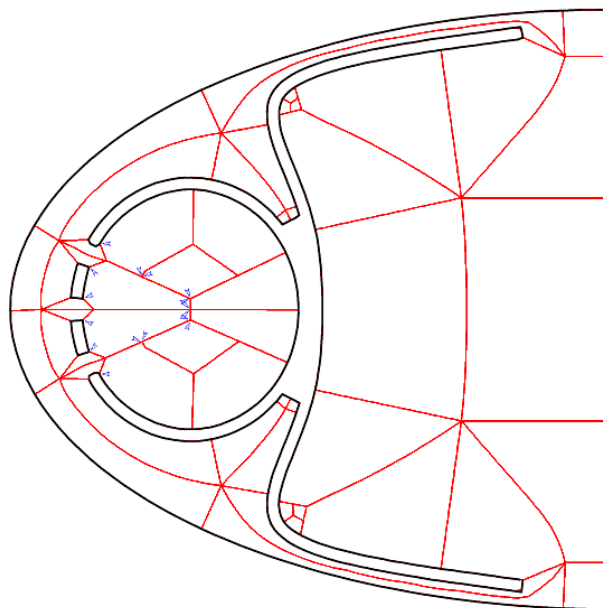


Figure 6: Final topology for aircraft wing anti-icing geometry

TopMaker is currently available for two-dimensional geometries, and extension to three-dimensions is underway. Since the geometries which currently require the most time for grid generation are three-dimensional, the three-dimensional version of TopMaker is where the greatest payoff for this work will occur. It is anticipated that the automation provided by the three-dimensional version of TopMaker, which is expected to be completed in 2005, will reduce the full grid generation cycle time by up to a factor of 10 for very complicated geometries. Most of the two-dimensional cases completed have required only a few seconds to run on an SGI Octane workstation.

SUMMARY AND CONCLUSIONS

The Glenn-HT code has been developed for solution of the three dimensional Navier-Stokes equations in turbulent flow in turbine geometries and other propulsion systems where convective heat transfer is important. The code has been validated for many turbine heat transfer problems, including tip heat transfer, external heat transfer with film cooling, and internal coolant passage heat transfer. Recently, the code has been applied to other aircraft problems such as aircraft wing anti-icing systems and, under the TBCC project, engine diffuser centerbody heat transfer.

In an effort to improve and expand the applicability of the predictive capability of Glenn-HT, as well as to improve the speed of problem solution, the Glenn-HT research effort has recently been directed in three primary areas – adding conjugate heat transfer capability to the code through the coupling of Glenn-HT with a boundary element method solver for the solid, improving the turbulence modeling capability of the code through implementation of anisotropic turbulence models, and incorporating automatic topology generation into the multi-block structured grid generation process. Work in these three areas is already providing improved code predictive capability and turn-around time.

Work in the coming year will continue in the areas described in this report, but will also begin to focus on testing of the unsteady capability of the code, ultimately leading to large eddy simulation (LES) capability. Use of the code will also be moved from operation on the SGI Unix platform to highly parallel linux-based clusters. Code development is also continuing on the development of a graphical user interface (GUI). As the code transitions from a research code to a more user-friendly application, the incorporation of these features will allow external users to more easily use the code for their convective heat transfer problems. The Glenn-HT code has been used by the domestic aircraft engine community, and is available for domestic use by request.

REFERENCES

- [1] Arnone, A., Liou, M.-S., and Povinelli, L. A., 1991, "Multigrid Calculation of Three-Dimensional Viscous Cascade Flows", AIAA Paper 91-3238.
- [2] Ameri, A. A. and Arnone, A., 1992, "Navier-Stokes Turbine Heat Transfer Predictions Using Two-Equation Turbulence Closures", AIAA Paper 92-3067 and NASA TM 105817.
- [3] Steinthorsson, E., Liou, M.-S., and Povinelli, L. A., 1993, "Development of an Explicit Multiblock/Multigrid Flow Solver for Viscous Flows in Complex Geometries", AIAA Paper 93-2380 and NASA TM 106356.
- [4] Ameri, A. A., Steinthorsson, E., and Rigby, D. L., 1999, "Effects of Tip Clearance and Casing Recess on Heat Transfer and Stage Efficiency in Axial Turbines", *Journal of Turbomachinery*, Vol. 121, pp. 683-693.
- [5] Ameri, A. A. and Bunker, R. S., 2000, "Heat Transfer and Flow on the First-Stage Blade Tip of a Power Generation Gas Turbine: Part 2 – Simulation Results", *Journal of Turbomachinery*, Vol. 122, pp. 272-277.
- [6] Garg, V. K. and Rigby, D. L., 1999, "Heat Transfer on a Film-Cooled Blade – Effect of Hole Physics", *International Journal of Heat and Fluid Flow*, Vol. 20, pp. 10-25.

- [7] Garg, V. K., 2000, "Heat Transfer on a Film-Cooled Rotating Blade", *International Journal of Heat and Fluid Flow*, Vol. 21, pp. 134-145.
- [8] Heidmann, J. D., Rigby, D. L., and Ameri, A. A., 2000, "A Three-Dimensional Coupled Internal/External Simulation of a Film-Cooled Turbine Vane", *Journal of Turbomachinery*, Vol. 122, pp. 348-359.
- [9] Rigby, D. L., Steinthorsson, E., and Ameri, A. A., 1997, "Numerical Prediction of Heat Transfer in a Channel with Ribs and Bleed", ASME Paper 97-GT-431.
- [10] Rigby, D. L., 1998, "Prediction of Heat and Mass Transfer in a Rotating Ribbed Coolant Passage with a 180 Degree Turn", ASME Paper 98-GT-329.
- [11] Kassab, A. J., Divo, E., Heidmann, J. D., Steinthorsson, E., and Rodriguez, F., 2003, "BEM/FVM Conjugate Heat Transfer Analysis of a Three-Dimensional Film Cooled Turbine Blade", *International Journal of Numerical Methods for Heat and Fluid Flow*, Vol. 13, No. 5, pp. 581-610.
- [12] Durbin, P.A. and Pettersson Reif, B.A., 2001, *Statistical Theory and Modeling for Turbulent Flows*, John Wiley & Sons LTD.
- [13] Rigby, D. L., 2003, "TopMaker: A Technique for Automatic Multi-Block Topology Generation Using the Medial Axis", *Proceedings of 4th ASME-JSME Joint Fluids Engineering Conference*, Honolulu, HI, FEDSM2003-45527.
- [14] Wilcox, D. C., 1998, *Turbulence Modeling for CFD, 2nd Edition*, DCW Industries, Inc., LaCanada, CA.
- [15] Wilcox, D. C., 1994, "Simulation of Transition with a Two-Equation Turbulence Model", *AIAA Journal*, Vol. 32, No. 2, pp. 247-255.
- [16] Menter, F. R., 1993, "Zonal Two-Equation $k-\omega$ Turbulence Models for Aerodynamic Flows", AIAA Paper 93-2906.
- [17] Chima, R. V., 1996, "A $k-\omega$ Turbulence Model for Quasi-Three-Dimensional Turbomachinery Flows", NASA TM 107051.
- [18] Schlichting, H., 1979, *Boundary Layer Theory*, 7th ed., McGraw-Hill, New York, pp. 312-313.
- [19] Garg, V. K., 2002, "Heat Transfer in Gas Turbines at NASA GRC", *International Journal of Heat and Fluid Flow*, Vol. 23, pp. 109-136.
- [20] Heidmann, J. D., Kassab, A. J., Divo, E. A., Rodriguez, F., and Steinthorsson, E., 2003, "Conjugate Heat Transfer Effects on a Realistic Film-Cooled Turbine Vane", ASME Paper GT2003-38553.
- [21] Giel, P. W., Van Fossen, G. J., Boyle, R. J., Thurman, D. R., and Civinskas, K. C., 1999, "Blade Heat Transfer Measurements and Predictions in a Transonic Turbine Cascade", ASME Paper 99-GT-125.
- [22] Menter, F. R., 1994, "Two-Equation Eddy Viscosity Turbulence Models for Engineering Applications", *AIAA Journal*, vol. 32, pp. 1598-1605.
- [23] Garg, V. K., Ameri, A. A., 2001, "Two-Equation Turbulence Models for Prediction of Heat Transfer on a Transonic Turbine Blade", ASME paper 2001-GT-0165.
- [24] Armstrong, C. G., Robinson, D. J., McKeag, R.M., Li, T.S., Bridgett, S.J., Donaghy, R.J., and McGleenan, C.A., 1995, "Medials for Meshing and More," 4th Int. Meshing Roundtable.
- [25] Program Development Corporation, 1997, "GridProTM/az3000 User's Guide and Reference Manual", White Plains, NY.